ANSYS Fluent Tutorial Part 2

1 Resuming Project

1. Open **Workbench**
2. **File**, **Open**, and ***Navigate*** to and select your **Workbench Project file** (**.wbpj**)
3. Your **Project Box** should return with the various checkmarks at the stages you have completed
4. To resume where you left off in Fluent (after the calculations were performed), click **Solution**

Note that if you select Setup, it will open Fluent to before the calculations were performed and you will need to recalculate.

Note that if you select Results, it will open a post-processing window that is not Fluent (not what we want).

2 Contours

One of the best ways to visually interpret your project and check that your answer seems reasonable is to view a contour plot of the solution. To do so, use the following procedure:

1. In the **Tree**, under **Results**, *select* **Graphics**
2. Within the **Task Page**, *select* **Contours** and then **Set Up**…
3. A Window of your **Contour options** will pop up
4. Under **Contours of**, change the dropdown menu to *select* **Velocity**…
5. Then select the sub-dropdown for **Velocity Magnitude**
6. Select **Display** and you will see a line contour plot of the Velocity Magnitude of your solution
7. Try *adjusting* the **Options** to interpret the contour better. *Check* the **Filled** box and *change* **Levels** to **100** and hit ***display*** again. Your Contour may show you some better flow details this way.
8. Become familiar with different Contours and evaluate if they're interesting to include in the report.

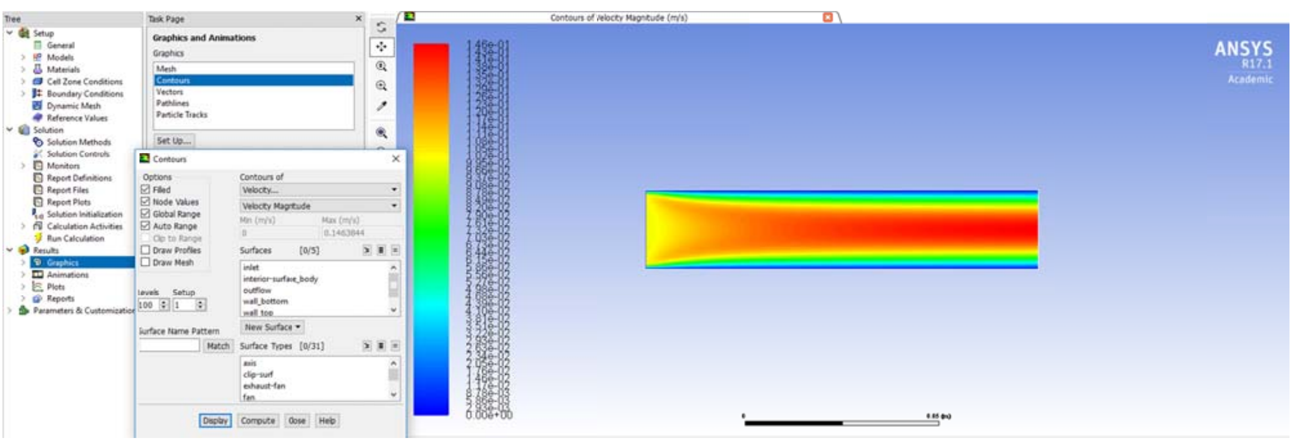

Figure 2.1

3 Saving an Image

To export the picture of the contour you just created, use the following procedure:

1. **File**, **Save Picture**…; A Window will pop up with your options.
2. Choose your **Format** (I recommend **<u>PNG</u>** for lossless image compression), **color**, and **resolution**.
3. If you choose the default **Resolution** (960 x 720) and save, your image will look like this:

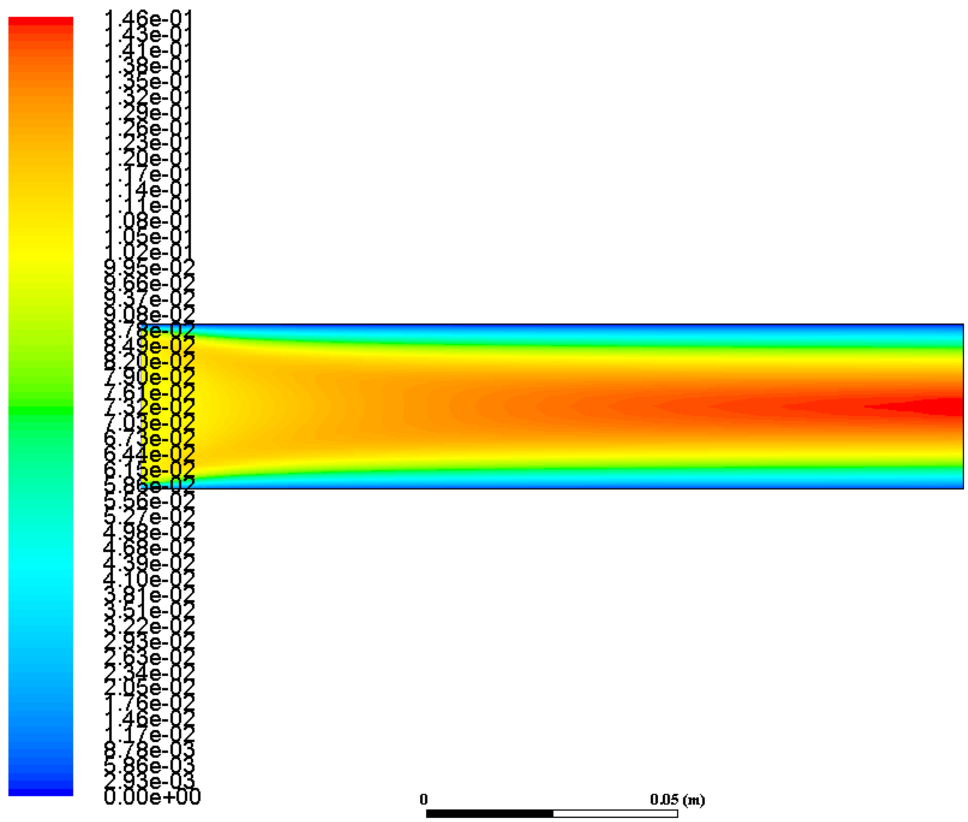

Figure 3.1

4. Notice that your color key is overlapping your geometry. Avoid this in your report.
5. A quick fix is to set your **<u>Resolution</u>** to **<u>1280</u>** x **<u>720</u>**. The increased width will center the geometry further away from the color key.
6. Alternately, you can adjust your **view** by **panning** and **zooming** out a bit with the **view buttons** on the immediate **left** of the **main window**. By changing the view on your screen, you change the view that saves in the image export.
7. You can **save** a **view** by selecting the **Viewing Tab** on the **top ribbon** and clicking **Views**… in the **Display category**.
8. Change the **Save Name** and click **Save**. Notice that another view was added to the list. At any point you can go back to this menu and select the new **View** and hit **Apply**.
9. Note that you will have to save several images to include in your report, so please get familiar.

4 Creating an XY Plot

You are asked to plot several features for your report. Here is an example on how to plot the **Outflow Velocity Profile**.

1. In the **Tree**, under **Results**, *select* **Plots**
2. Within the **Task Page**, *select* **XY Plot** and then **Set Up**…
3. Options
    a. Node Values – **<u>Check this box</u>** if you want the data from your mesh to be from the cell edges (nodes) or the center of the cell (if box is unchecked).
    b. **<u>Position on X Axis</u>** – Check this box if you want your independent variable on the x-axis.
    c. Position on Y-Axis – Check this box if you want your independent variable on the y-axis
    d. Write to File – When you're ready to export your XY plot data, select this box
4. Plot Direction – Plot your dependent variable as a function of x or $\underline{\boldsymbol{y}}$.
5. Y Axis Function – To plot the Outflow Velocity Profile, select **<u>Velocity</u>**, then **<u>Velocity Magnitude</u>**.
6. Surfaces – Select the **<u>Outflow</u>** to plot the velocity profile as a function of y.
7. Click Plot. Your window should look like this:

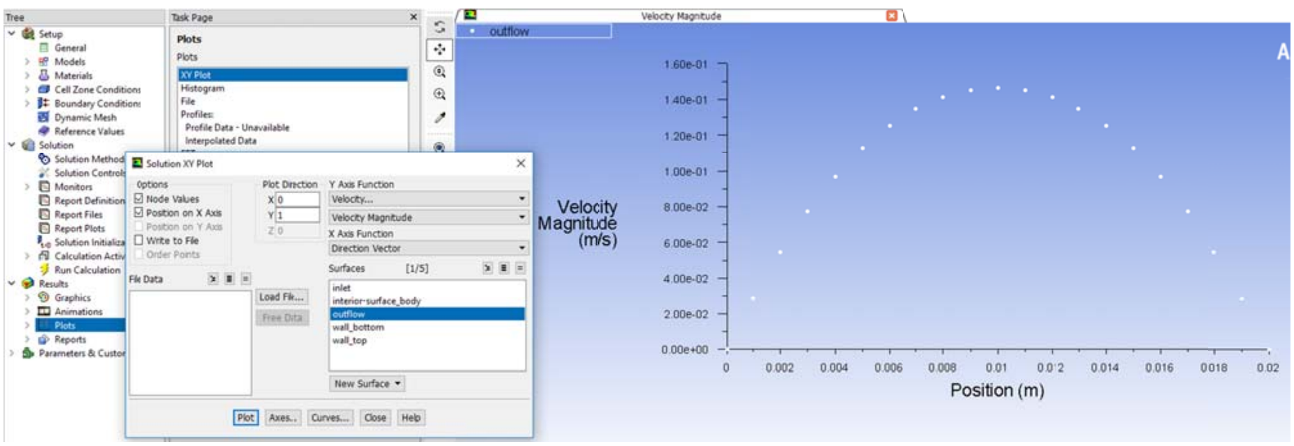

Figure 4.1

8. You can save this picture by following the procedure above.
9. To Export the data, ***Check the Box*** **Write to File**. Before Exporting the Data, it's a good idea to plot it first and make sure the axes and data are correct.
10. Select the location and file name. It will be a text file. You can open and view the data in notepad.
11. You'll be able to Import this data into Matlab for plotting. This is necessary when you need to directly compare (on the same figure) data from 2 different Fluent simulations and theory (see your project 1 checklist). There is a Matlab code provided to you later in this Tutorial.

To plot the Wall Shear Stress along the wall, repeat the procedure above, but select **<u>Wall Fluxes</u>** and **<u>Wall Shear Stress</u>** under the **Y Axis Function**. You'll also need to select the correct **Surface** of the **bottom wall** instead of the Outflow. And note that you'll be plotting **Wall Shear Stress** as a function of $\underline{\boldsymbol{x}}$ instead, so you'll need to adjust the **Plot Direction** respectively.

5 Wall Skin Friction

The Project asks you to make a table of the wall skin friction coefficients for multiple fluid velocities using multiple methods. It asks for the Fluent calculation, classical theory calculations, and reading off the Moody Diagram.

5.1 Fluent

To plot the Skin Friction Coefficient along the wall, repeat the procedure in Creating an XY plot, but select **Wall Fluxes** and **Skin Friction Coefficient** under the **Y Axis Function**. You'll also need to select the correct **Surface** of the **bottom wall** instead of the Outflow. And note that you'll be plotting the **Friction Coefficient** as a function of ***x*** instead, so you'll need to adjust the **Plot Direction** respectively. Note that the Friction Coefficient below converges to 1e-3. This is the value to include in your table.

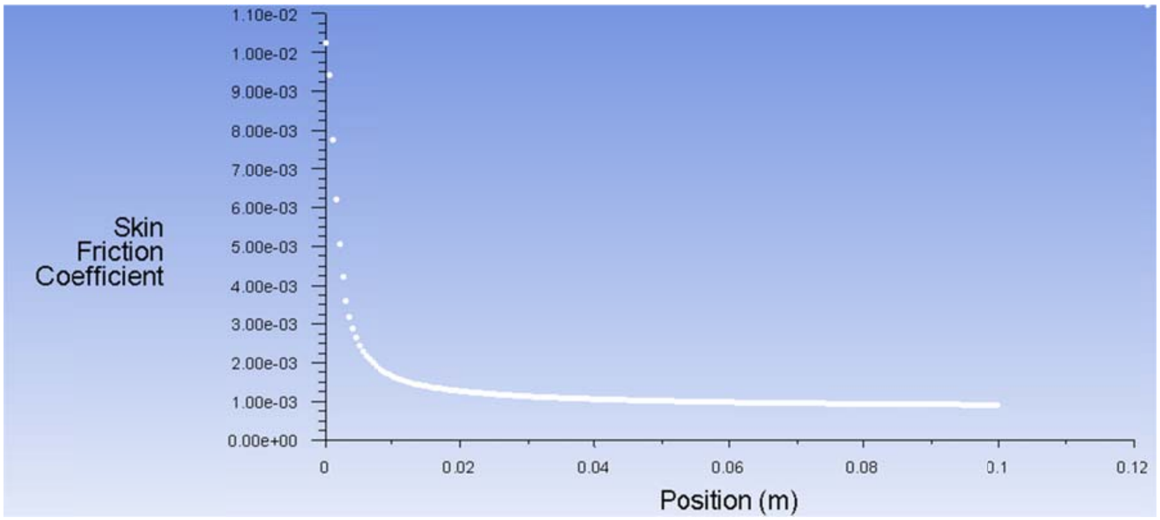

Figure 5.1

5.2 Theory

For Laminar Flow (Boundary Layer)

$$C_f = \frac{0.664}{\sqrt{Re_x}}$$

$$Re_x = \frac{\rho \bar{u} x}{\mu}$$

For Turbulent Flow (boundary Layer)

$$C_f = \frac{0.455}{\log(Re_x)^{2.58}}$$

6 Creating a Rake

A Rake is a line of sample points. Right now, you've been able to plot the velocity profile at the **Outflow** because you were able to select the **Surface**. But what if you wanted to plot the velocity as a **function** of y at $x = 0.05\ m$? You can create a line to interpolate data points at $x = 0.05$ m from $y = 0 - 0.02\ m$.

1. In the **XY Plot** window (same as before), under **Surfaces**, select **New Surface** at the bottom
2. Select **Line**/**Rake**. A window of **Line**/**Rake Options** will pop up.
3. Change Type from **Line** to **Rake**
4. Set the **Number of Points** to the number of data points you want to take across the y-axis. Note that your mesh is currently 1 mm spacing for a 2 cm high geometry, so you currently have **20 nodes**. Selecting this value depends on the resolution of the data you're trying to capture. For this case (velocity profile), ***20 points is fine***. However, in Project 2 for Boundary Layer flow, you may need to capture a much higher resolution of 100+ points.
5. Select your **start** and **end points**. A rake at $x = 0.05$ from $y = 0 - 0.02$ would result in points at $x_0 = 0.05, y_0 = 0, x_1 = 0.05, y_1 = 0.02$.
6. Name the rake something descriptive, such as, **x=0.05**
7. This rake will now show up in your **Surfaces list** in the **XY plot menu**. Plot along this new rake.
8. You'll notice from the plot below that you can plot multiple curves simultaneously by selecting multiple Surfaces. This plot shows the developing velocity profile at $x = 0.02$, $x = 0.05$, and $x = 0.10\ m$. Note that this is one of your Project 1 objectives. Boost your street cred by plotting this in Matlab.

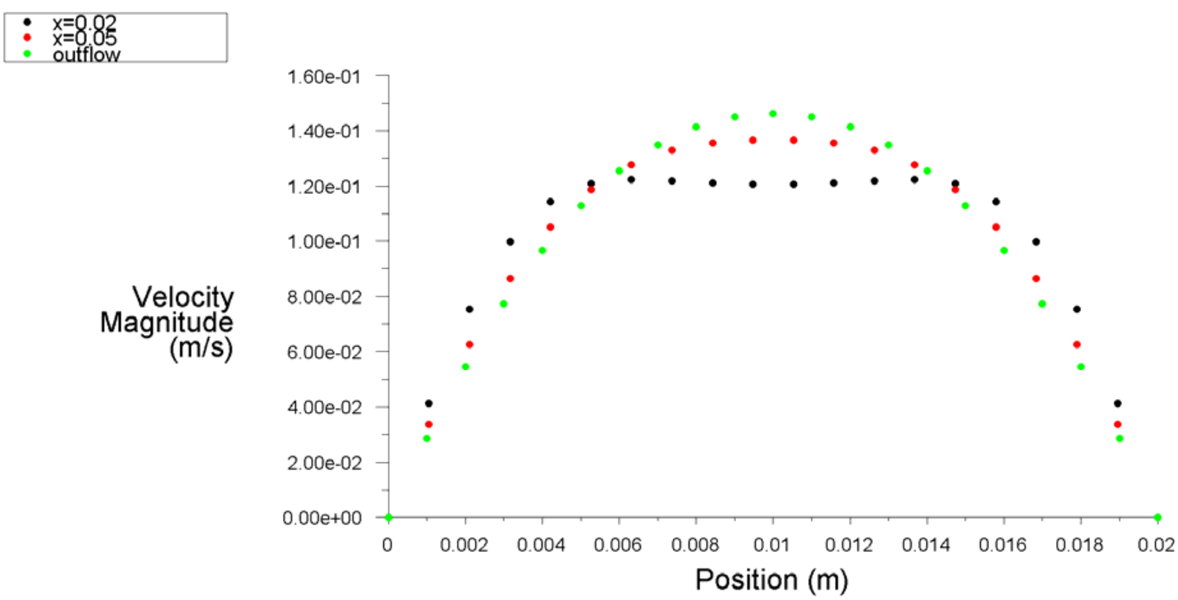

Figure 6.1

7 Importing XY Plot Data to Matlab

When plotting an XY Plot, check the Write to File box to export the data points.

Copy this code into Matlab and fill in the Inputs to match your conditions.

```
%% readFluentXYdata.m
% Read the XY data from Fluent
% ME 326 - Spring 2017
% Matthew Brown
% 02/04/17

close all; clear all; clc;
home = pwd;

%% Inputs – Edit these values as necessary
fileLocation = 'E:\Google Drive\Clarkson\ME 326 F16\Project1';
fileName = 'file-name';
xDataLabel = 'x (m)';
yDataLabel = 'U (m/s)';


% Figure properties
fontpt = 14;
savename = 'P01_velocityProfile';
SAVE = 0; % Change to 1 to save the image of your plot in current directory


%% Import data

cd(fileLocation)
fluent = importdata(fileName);
cd(home)
x = fluent.data(:,1);
y = fluent.data(:,2);

%% Plots

fig1 = figure;
plot(x,y)
xlabel(xDataLabel)
ylabel(yDataLabel)
set(gca,'fontsize',fontpt)

if SAVE == 1;
   print(fig1, savename, '-dpng', '-r600')
end
```

8 Refining the Mesh

Every simulation you run, you should be asking yourself an important question: "How do I know that my mesh is fine enough to give me an accurate solution?" To answer this, refine your mesh to double to resolution and recalculate the solution. If your solution doesn't change, your initial mesh was satisfactory.

1. **Save your work**, close **Fluent**, and view your **Project** in **Ansys Workbench**
2. On your **Project**, select the **dropdown arrow** (top left corner of the project) and select **Duplicate**
3. This will make a copy of your Project. Rename it ***Project 1 Laminar Flow Fine Mesh***.
4. The reason you made a copy is so you don't lose your coarse mesh data in case you need to go back to grab some images or data for your report.
5. *Double click* **setup** on your **new project** (**fine mesh**) to re-enter **Fluent**.

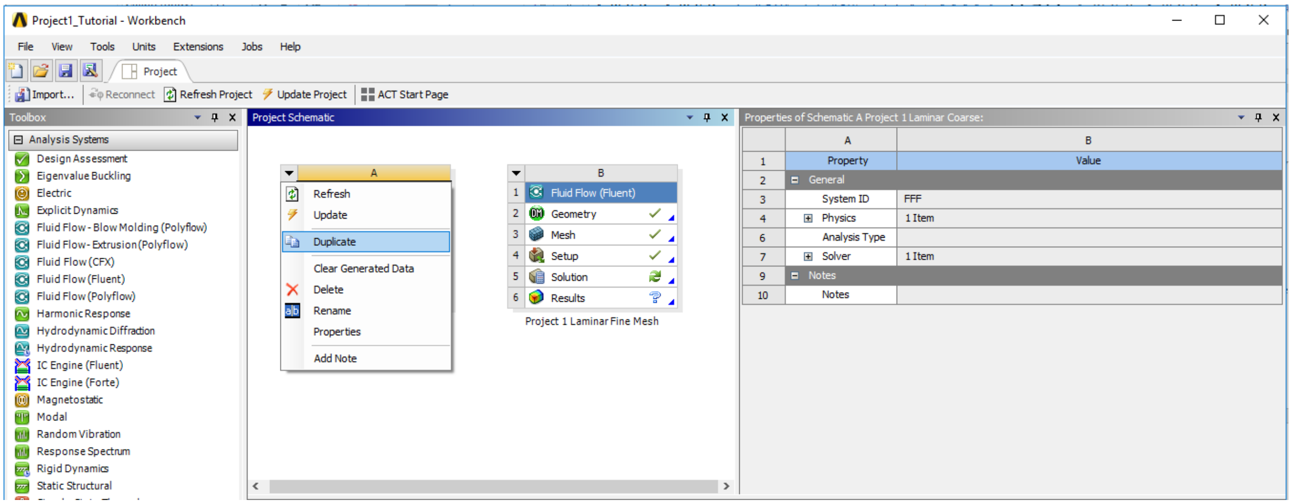

Figure 8.1

6. Once **Fluent** opens, select **General** in the **Tree** menu and **Mesh Display** in the **task page**.
7. Select **<u>all surfaces</u>** and click **Display**. DO NOT CLOSE THIS WINDOW. Just move it to the side.
8. In the **top Ribbon menu**, under **Setting up the Domain** (first tab), there is a section called **Adapt**. In this section, click the dropdown **Mark/Adapt Cells** and select **Region**
9. Since you want to **adapt** the whole geometry, select the options for **<u>Inside</u>** a **<u>Quad</u>** and set the coordinates for the lower left corner **<u>(0,0)</u>** and the upper right corner of your geometry **<u>(0.1, 0.02)</u>**.
10. Click **Adapt** ONE TIME. There will be a warning message saying your mesh refinement won't last after this session. Click OK. Note that when you close and open fluent again, the refinement is lost.
11. Now in your **Mesh Display Window** that you set aside before, *click* **Display** again. You should see your Mesh resolution has doubled.
12. Re-run your simulation procedure (from Tutorial 1). You will need to re-initialize and re-calculate. Make sure your Boundary Conditions are still correct.
13. Use the previous tutorials to obtain the proper plots from the solution to include in your report (see Project 1 Checklist).
14. You may need to export the outflow velocity to compare with before your mesh refinement.

9 Turbulent Simulation

1. **Save** your work, close **Fluent**, and view your **Project** in **Ansys Workbench**
2. On your **refined mesh Project**, **Duplicate** it (same as before)
3. **Rename** the copy ***Project 1 Turbulent Flow Fine Mesh***.
4. *Double click* **setup** on your new project (fine mesh) to re-enter **Fluent**.
5. If your **Mesh refinement** didn't carry over (it won't), I recommend **refining** it.
6. In the **Tree menu** under **Models**, edit **Viscous**-**Laminar Model** and change it to **k-epsilon** (**2-eqn**)
7. Keep all **<u>default values</u>** and *click* **OK**.
8. Now *edit* your **Boundary Conditions** to correspond to the new **Turbulent velocity** of **5 m/s**
9. In the **Boundary Conditions** tab, select your **Velocity Inlet** and ***Edit*** the **Properties**
10. The Velocity Magnitude is 5 m/s
11. You'll notice a new **Turbulence section** appeared in the **properties**. Change the **Specification Method** to read **<u>Turbulence Intensity and Hydraulic Diameter</u>**. Plug in the following values.
12. The **Turbulence Intensity** at the Core of fully-developed duct flow is estimated as:

$$I = 0.16Re^{-1/8}$$

$$Re = \frac{\rho \bar{u} D_H}{\mu}$$

For a rectangular Duct,

$$D_H = \frac{4ab}{2(a+b)} = \frac{2a}{1+\frac{a}{b}}$$

Here a is the duct height and b is the width (depth into the page). As b approaches infinity (as in our case),

$$D_H = 2a$$

13. Continue down the **Tree** to **Solution Methods**. Try changing the **Turbulent Kinetic Energy** and **Dissipation Rate** to **<u>Second Order Upwind</u>**. If the solution has trouble converging later, you can come back to this tab to relax these conditions to **First Order**.
14. **Initialize** the **Solution** from the **inlet conditions** again.
15. **Calculate**. You may need to *run more calculations* until the residuals level off.
16. Proceed getting the proper Plots and Figures for the Project

10 Displaying Results

You need to think about what is the best way to display your results. Here is an example of multiple methods showing the same Data of comparing velocity profiles for Laminar and Turbulent flows. Which method makes the clearest and most concise statement?

a)

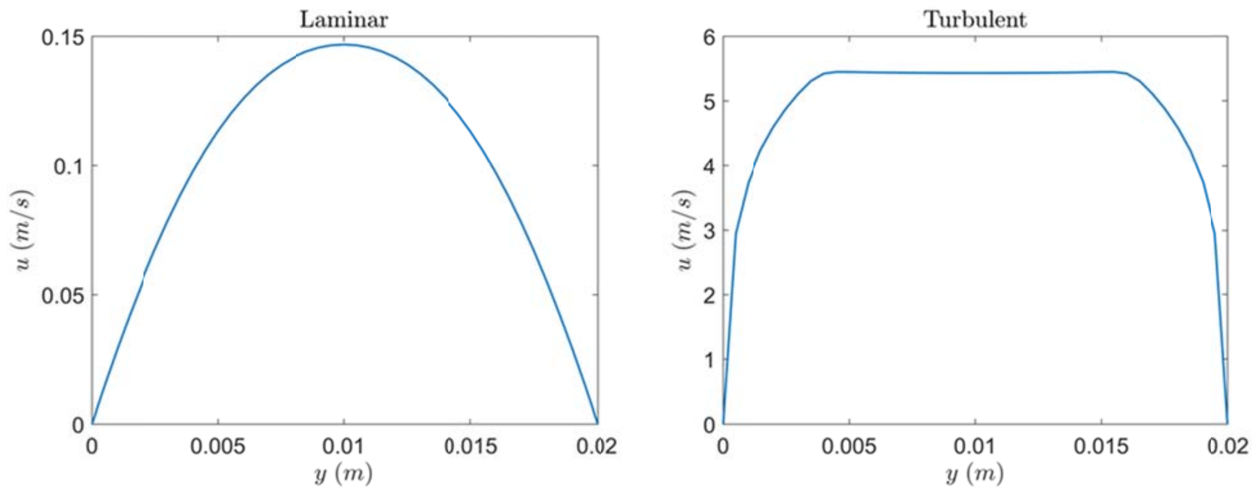

Figure 10.1

b)

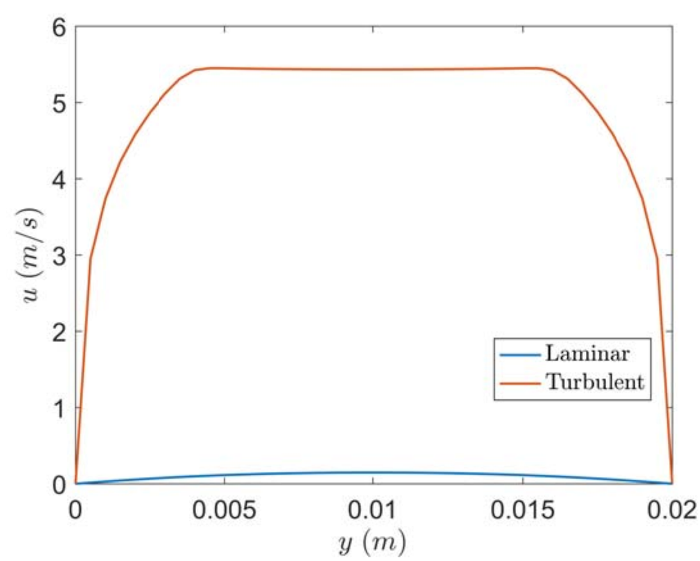

Figure 10.2

c)

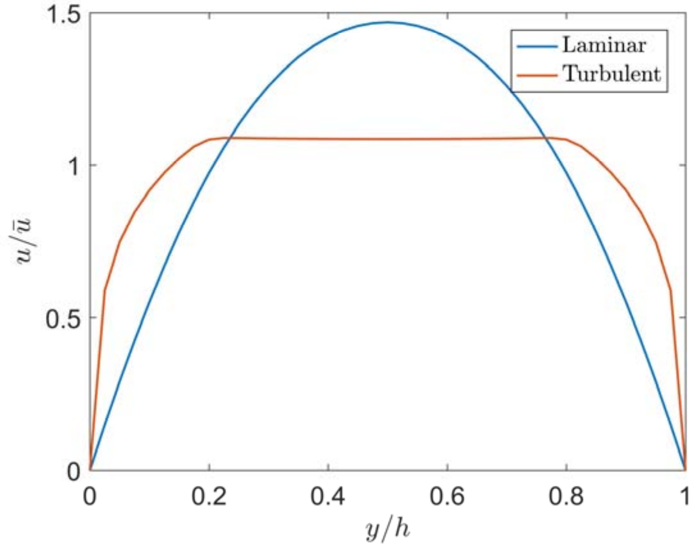

Figure 10.3